PRESTRESSED MODAL ANALYSIS OF NON UNIFORM BEAM USING ANSYS

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF
THE REQUIREMENTS FOR THE DEGREE OF

Bachelor of Technology

in

Mechanical Engineering

By

Mohammed Istiyaq and **Ayush Agarwal**

(ROLL NO.: 110ME0294) (ROLL NO.:110ME0278)

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA-769008

ACKNOWLEDGEMENT

Successful completion of work will never be one man's task. It requires hard work in right direction. There are many who have helped to make my experience as a student a rewarding one. In particular, I express my gratitude and deep regards to my thesis guide **Prof. Anirban Mitra** first for his valuable guidance, constant encouragement and kind co-operation throughout the period of work which has been instrumental in the success of thesis. I also express my sincere gratitude to **Prof. K.P. Maity**, Head of the Department, Mechanical Engineering, for providing valuable departmental facilities. I would also like to thank my fellow students.

Mohammed Istiyaq

110ME0294

Ayush Agarwal

110ME0278

Mechanical Engineering

National Institute of Technology, Rourkela

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

CERTIFICATE

This is to certify that the thesis " Pre-Stressed modal analysis of non-uniform beam in Ansys ", submitted by Ayush Agarwal and Mohammed Istiyaq, B. Tech students of National Institute of Technology in Mechanical department, is a genuine work done by them under my guidance. To the best of my knowledge, the dissertation is entirely their own work. The work contained in this thesis has not been submitted to any other university/ institute for award of any Degree or Diploma.

Prof. Anirban Mitra

Dept. of Mechanical Engineering

National Institute of Technology,

Rourkela-769008

CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

Beams are widely used structural members and its dynamic characteristics under loading is of great importance and vital for study. Thus, the purpose of the present thesis paper is to carry out an efficient and accurate simulation for prestressed modal analysis of non-uniform beam using Ansys. Geometric non-linearity is also considered. Displacement and frequencies of cantilever beam are studied with distributed as well as point load. The result thus obtained was verified with the result as obtained as the output of the programme in other literature survey.

Chapter 1 include general introduction about the topic and related research in non-linearity in vibration of beams. Chapter 2 deals in literature review by help of which this work has been completed with satisfactory result. The theoretical background of the topic including the use of beam, relevant pre-stressed condition and its modal analysis is discussed in chapter 3.The methodology adopted is given in chapter 4.Chapter 5 deals with step by step Procedure in Ansys. Chapter 6 sums up the work with result and conclusion. Some of the publications that were referred are mentioned in Chapter 7.

CHAPTER 1

GENERAL INTRODUCTION

GENERAL INTRODUCTION

1.1 INTRODUCTION AND BACKGROUND

Beams have always been used extensively in building materials and structures. Depending on the application and working environment they are frequently under dynamic loading condition. Hence, from the standpoint of proper functioning and safety, study of response of these structural elements under dynamic loading is of utmost importance.

Dynamic analysis of a structure or structural element consists of two different aspects, namely, free and forced vibration studies. The main purpose of a free vibration analysis is to predict the natural frequencies of various vibration modes of the component under no load and loaded conditions. The natural frequencies of the structure under some form of static loading are generally known as loaded natural frequencies. Determination of these frequencies is important to avoid resonance condition, where the external excitation frequency coincides with the system natural frequencies. At resonance condition, the element vibrates with large amplitude resulting in deterioration of structural health. This deterioration usually causes local increase in flexibility, which is a serious threat to life and performance of structural component. Thus in last two decades the study and analysis of natural frequencies in normal and loaded conditions has become increasingly important. From design point of view, maximum vibration amplitude must be limited to a suitable amount for a structure to perform safely.

Non- prismatic beams also find wide-spread use in engineering fields, such as, aerospace, marine, construction engineering etc. These are beams of varying or abruptly decreasing cross-section which find huge application in weight and strength optimisation of structure. They are also used in increasing aesthetic side of a structure in design engineering. The study of dynamic properties of a structure under vibrational excitation is called Modal Analysis.

1.2 NON LINEAR ANALYSIS

It is well known that nonlinearities are inherent in mechanical systems and the two most commonly encountered types of nonlinearities are geometric nonlinearity and material nonlinearity. Nonlinear strain-displacement relations give rise to geometric nonlinearity, while material nonlinearity is associated with nonlinear stress-strain relations. Depending on the nature of the problem one or both of the nonlinearities can be included in the analysis. Yet, in the earlier years, mechanical designs were based on linear models in order to simplify the analysis. However, improvement in the field of computational powers of the modern computers has changed the scenario.

Normally in any problem solving it is assumed that displacements are infinitesimally small compared to the length of the structure and material is linearly elastic. It is also assumed that the nature of the boundary conditions applied on the structure is constant and never change. With all these assumptions, the relation of the displacement thus obtained is linear with the load applied to the structure. If the displacement and load applied does not follow linear relationship then it is called a non-linear function and a non-linear analysis is required to predict the behaviour. For example, in case of a static problem, if the deflection under loading becomes comparable to the thickness (relevant lateral dimension) of an element it becomes a large displacement problem/analysis. The phrase 'large displacement' (or in case of a dynamic analysis ‘large amplitude’) is exclusively associated with geometric nonlinearity. In such situations deflections of the structure are large compared to the original dimensions and the stiffness of the system changes.

1.3 WHY MODAL ANALYSIS?

One of the uses of modal analysis is to determine the vibration response characteristics (natural frequencies and mode shapes) that a structural element or the machine component shows during run time, while it is being designed. It is also a starting point for further study of another, more detailed, dynamic analysis, such as a transient dynamic analysis or a harmonic response analysis and a spectrum analysis. Modal analysis is also often used to determine the natural frequency and mode shape of a structure. The natural frequencies and mode shapes play a vital role in the design of a structure for dynamic loading conditions.

1.4 VIBRATION AND FREQUENCY

Zero-value displacement is the only load that is valid in a typical modal analysis constraint. If a nonzero displacement constraint is given as input, the program will assign the DOF with a zero-value constraint. Other loads are ignored even if they are specified.

Natural Frequency: All structures have a natural frequency with which they oscillate. If a structure under normal condition is subjected to an excitation which is close to its natural frequency, the structure starts oscillating with large amplitude than in normal condition. By studying the results obtained from a modal analysis it can be ascertained whether a model requires either more or less damping for it to prevent failures in future in its life. Modal analysis can also be used to find out the frequency at which resonance will occur, and thus modify the design of the structure, under specific constraints.

Modes of Vibration and Mode shapes: A structural element is a continuous system and can vibrate in many different ways. These different ways of vibration each have their own frequency, which is determined by moving mass in that mode, and the restoring force that tries to return that specific distortion of the body back to its equilibrium position. Each of these different ways of vibrating is known as a mode of vibration of the system. Each mode is assigned a number and the lowest frequency at which a system vibrates after all external loads are removed is assigned as mode 1 or fundamental mode.

A mode of vibration is characterized by a modal frequency and a mode shape. Each mode is entirely independent of all other modes. Thus all modes have different frequencies and different mode shapes. In the study of vibration in engineering, the expected shape/curvature (or displacement) of a system at a particular mode due to vibration is the mode shape. Thus the mode shape always describes the time-to-time curvature of the system under vibration where the magnitude of the curvature continuously changes. The mode shape depends on two factors:

1) Shape (Geometrical parameters) of the system

2) Boundary conditions of the system.

CHAPTER 2

LITERATURE REVIEW

LITERATURE REVIEW

Different researchers have carried out various theoretical and experimental studies to investigate several aspects of uniform as well as non-uniform beams. They have employed different approaches and methodologies to analyse such structural elements. The following section briefly describes a few of the literatures available related to research work on beams.

M. Baghani, H. Mazaheri and H. Salarieh [1] have presented analytical expression for large amplitude free vibration of beam. They have also considered geometric non-linearity and have solved the expression using Variational Iteration Method (VIM). Also the different nonlinear frequencies have been considered for different shapes of modes.

Allahverdizadeh, I. Eshraghi, M.J. Mahjoob and N. Nasrollahzadeh [2] have studied the variation of characteristics that are dependent on the amplitude of the functionally graded sandwich beams. The nonlinear characteristic of the material is also considered by applying it with the help of an exponential function. Also along with taking all this into consideration they have derived the governing equation of free vibration by means of finite element method.

H. Arvin and F. Bakhtiari-Nejad [3] have used Von Karman strain displacement equation. Nonlinear equations are obtained by Hamilton equation. Non-linear normal mode frequencies and non-linear natural frequencies are observed with and without natural frequencies. In case of the internal resonance, the internal resonance between the two transverse modes and between one transverse and one axial mode are explored.

Lianhua Wang, Jianjun Ma, Jian Peng and Lifeng Li [4] have investigated the nonlinear vibrations and the parametric instability of the structure on the elastic foundation. The motion equation is obtained by the Hamilton equation considering the inextension. The discrete modal is obtained by Galerkin equation.

M. Baghani, R.A. Jafari-Talookolaei and H. Salarieh [5] have presented analytical expressions for the large amplitude free vibration and also have analysed post buckling of laminated beams on elastic foundation. Von – Karman equation is considered for obtaining the geometric nonlinearity. In this third order approximation of VIM is done thus getting more accurate results.

Carlos E.N. Mazzilli, César T. Sanches, Odulpho G.P. BarachoNeto, Marian Wiercigroch and Marko Keber [6] have derived nonlinear equation governing the dynamics of loaded beam where load is given axially. Attempts have been done to make a low-dimensional modal. Two states were considered regarding the application of force, thrust force and uniformly distributed load.

Sandro Carbonari, Fabrizio Gara, Davide Roia, Graziano Leoni and Luigino Dezi [7] have done the experimental tests that were carried out on two pre-stressed thin walled V-shaped elements. After the experiments the modal shapes and the free vibration were noted. Then the combination of the dynamic experiment and the ultimate static load tests allowed finite element model calibration accurately, which accounted for both the mechanical and the geometrical nonlinearities, capable to predict the behaviour of the elements up to collapse and useful to design repairing and strengthening works.

T. Kocatürk, M. Şimşek [8] have studied the dynamic response of pre-stressed beam under harmonic loads. The constraints are considered using the Lagrange multiplier. The effects of the value of eccentricity of compressive load and the frequency are studied in detail.

CHAPTER 3

ANALYSIS IN ANSYS

3.1 Procedure for performing modal analysis on a cantilever beam

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Solid ->10 node 187 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (210e09,0.3 respectively).

4. Material Properties ->Material Models ->select favourites->density (density=7850 was fed)

5. Modeling ->Create -> Areas -> rectangular -> By dimensions(breadth=0.002,height=0.005)

6. Modeling->Operate->extrude ->Areas->along Normal (required dimension were fed i.e 1m)

7. Meshing ->Mesh Tool (smart size and the shape of the mesh were chosen and then the
structure was meshed.The element edge length was specified as 0.009)

8. Go to Solution ->analysis Type ->New analysis->select Modal

9. Go to Solution ->analysis Type ->Analysis option->select “Block Lanczos”

Next set the no. of modes to extract to 5 next click on “expand mode shapes” ,

next set 5 to no. of modes to expand and

set the frequency range from 0 to 10000.

10.Go to solution->Define loads->apply->structural ->displacement->on areas Select the area to be fixed by clicking over it and set the displacement value to zero.

11.Next go to Solution->Solve->current LS->click OK to execute the solution and click close on the dialogue box when solution is done.

12.Generalpostproc->Result summary(list of all nodal frequencies are displayed)

13.Generalpostproc->Read Result->By Pick(picking the last nodal frequency)

14. General postproc->Plot Result->Deformed Shape->click on “deformed+undeformedshape”we get this shape

3.2 Procedure for performing modal analysis on a Clamped-Clamped beam

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Solid ->10 node 187 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (210e09,0.3 respectively).

4. Material Properties ->Material Models ->select favourites->density(density=7850 was fed)

5. Modeling ->Create -> Areas -> rectangular -> By dimensions(breadth=0.002,height=0.005)

6. Modeling->Operate->extrude ->Areas->along Normal (required dimension were fed i.e 1m)

7. Meshing ->Mesh Tool (smart size and the shape of the mesh were chosen and then the
structure was meshed.The element edge length was specified as 0.009)

8. Go to Solution ->analysis Type ->New analysis->select Modal

9. Go to Solution ->analysis Type ->Analysis option->select "Block Lanczos"

Next set the no. of modes to extract to 5

next click on "expand mode shapes" ,

next set 5 to no. of modes to expand and

set the frequency range from 0 to 10000.

10.Go to solution->Define loads->apply->structural ->displacement->on areas
Select both the areas to be fixed by clicking over it and set the displacement value to zero.
11.Next go to Solution->Solve->current LS->click OK to execute the solution and click close on the dialogue box when solution is done.
12.Generalpostproc->Result summary(list of all nodal frequencies are displayed)

13.Generalpostproc->Read Result->By Pick(picking the last nodal frequency)

14. General postproc->Plot Result->Deformed Shape->click on "deformed+undeformedshape"

3.3 Procedure for performing modal analysis on a Tapered Cantilever beam

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Beam->2 Node 188 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (2e05,0.3 respectively).

4.Preprocessor->Sections->Beams->Common sections->define breadth and height of the section(B=40, H=20)->Name it as small->Apply

5.Change ID and Name to 2 and Big->Change the dimension to the larger end of the tapered beam(B=80,H=20)->Click Apply->OK

6. Select Preprocessor->Sections->Beams->Tapered sections->By XYZ Location->Name the section as tapered and change the X location equal to length of the beam(1000mm)

7.Preprocessor->Modelling->create->Keypoints->In Active CS->

Create 2 keypoints

Keypoints	Coordinates(x,y)
1	(0,0)

2	(1000,0)

8. Go to Preprocessor->Modelling->create->Lines->Lines->Straight Lines->click on the two keypoints and OK.

9. Go to Preprocessor->Meshing->size control->Manual size->Global->size->set no. of element division to 50

10. Again Go to Preprocessor->Meshing->Mesh Attributes->Default attribs->set section no. to 3 tapered.

11. Preprocessor->Meshing->Mesh->Lines->click on line and OK

12.PlotControl->style->Size and shape->ON the display of element

13. Go to Solution ->analysis Type ->New analysis->select Modal

14. Go to Solution ->analysis Type ->Analysis option->select “Block Lanczos”

Next set the no. of modes to extract to 5

next click on “expand mode shapes” ,

next set 5 to no. of modes to expand and

set the frequency range from 0 to 10000.

15.Go to solution->Define loads->apply->structural ->displacement->on Keypoint

Select keypoint 1 to be fixed by clicking over it and set the displacement value to zero.

16.Solution->Solve->Current LS

17. Generalpostproc->Result summary(list of all nodal frequencies are displayed)

18. Generalpostproc->Read Result->By Pick(picking the last nodal frequency)

19. General postproc->Plot Result->Deformed Shape->click on "deformed+undeformedshape"

Procedure for performing non-linear static analysis on a Cantilever beam (for Point Load)

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Beam ->2 node 188 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (74.76e09,0.33 respectively).

4. Material Properties ->Material Models ->select favourites->density(density=2800 was fed)

5. Preprocessor->Modelling->Create->Keypoint->In Active CS

Two Keypoints were defined for this structure to create a beam with a length of 0.406m

Keypoints	Coordinates(x,y)
1	(0,0)
2	(0.406,0)

6. Preprocessor->Modelling->Create->lines->lines->Straight Line

A Line was created between keypoint 1 and keypoint 2

7. Preprocessor> Meshing > Size Cntrls>ManualSize> Lines > All Lines...

(50 element divisions along the line).

8.Preprocessor> Meshing > Mesh > Lines > click 'Pick All'

9. Go to Solution > New Analysis > Static

10. Select Solution > Analysis Type >Sol'n Control...

The following values were fed as shown in fig.

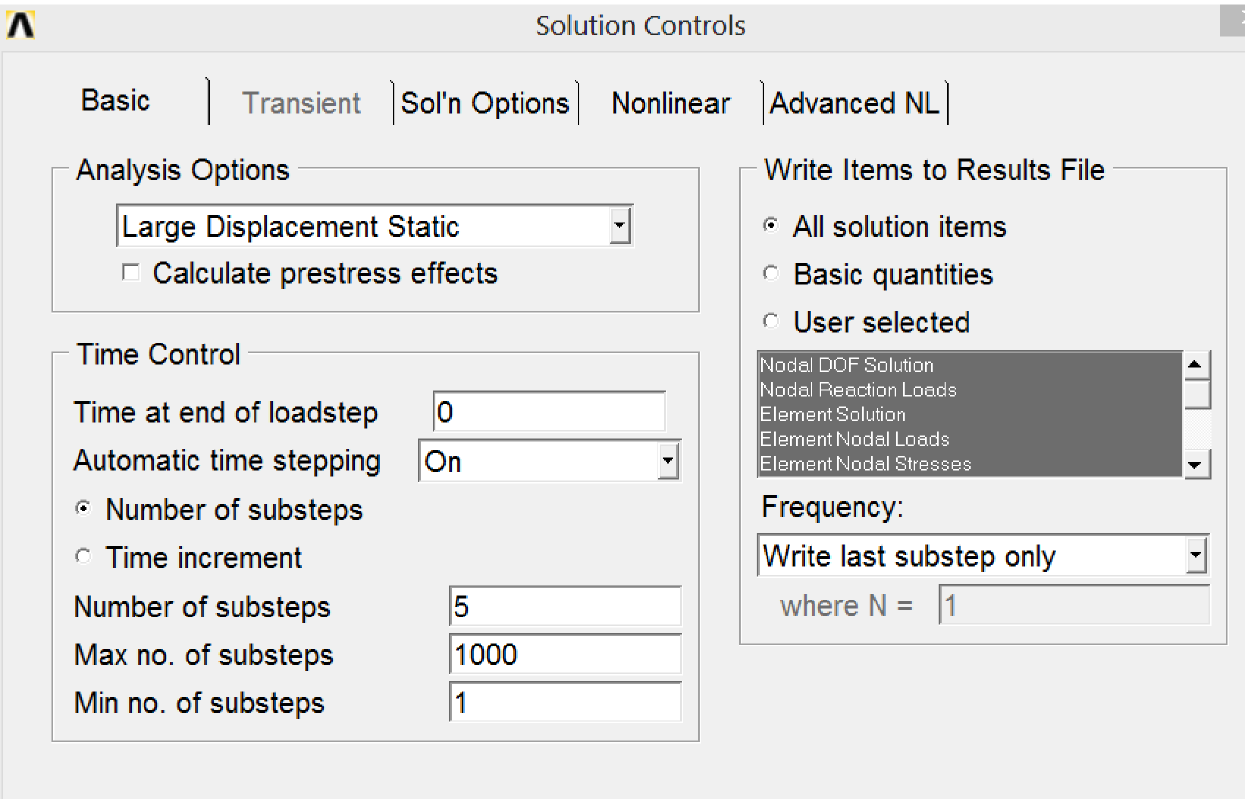

Fig.3.1 Values that are fed in Sol'n and Control for non-linear analysis of Cantilever Beam

11.Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Keypoint 1 was fixed by making the displacement value to zero (ie all DOFs constrained).

12.Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Place different values of loads on the right side of the beam i.eKeypoint 2.

13.Solution > Solve > Current LS

14.General Postproc> Plot Results > Deformed Shape... >Def + undeformed

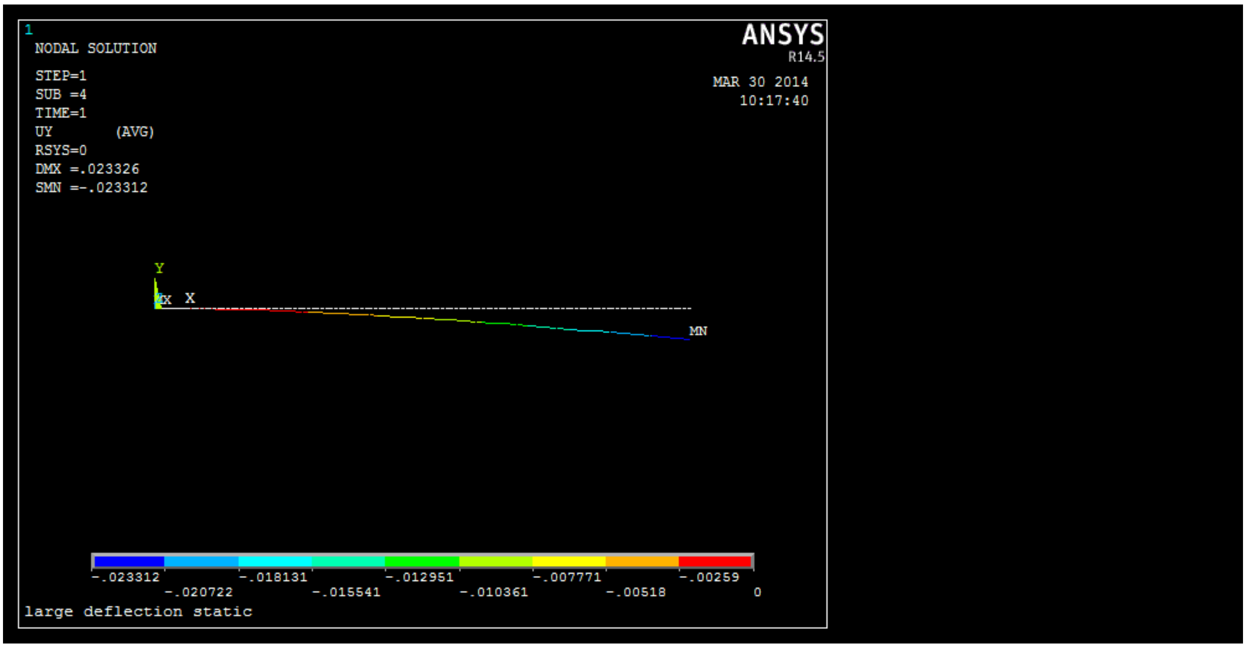

Fig. 3.2 Deflection Profile of Cantilever Beam for Non-Linear Analysis

Procedure for performing non-linear static analysis on a Clamped-Clamped beam (for Point Load)

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Beam ->2 node 188 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (210e09,0.3 respectively).

4. Material Properties ->Material Models ->select favourites->density(density=2800 was fed)

5. Preprocessor->Modelling->Create->Keypoint->In Active CS

Two Keypoints were defined for this structure to create a beam with a length of 0.406m

Keypoints	Coordinates(x,y)
1	(0,0)
2	(0.5,0)
3	(1,0)

6. Preprocessor->Modelling->Create->lines->lines->Straight Line

A Line was created in between keypoint 1 and keypoint 3 including Keypoint 2

7. Preprocessor> Meshing > Size Cntrls>ManualSize> Lines > All Lines...

(50 element divisions along the line).

8.Preprocessor> Meshing > Mesh > Lines > click 'Pick All'

9. Go to Solution > New Analysis > Static

10. Select Solution > Analysis Type >Sol'n Control...

The following values were fed as shown in fig.

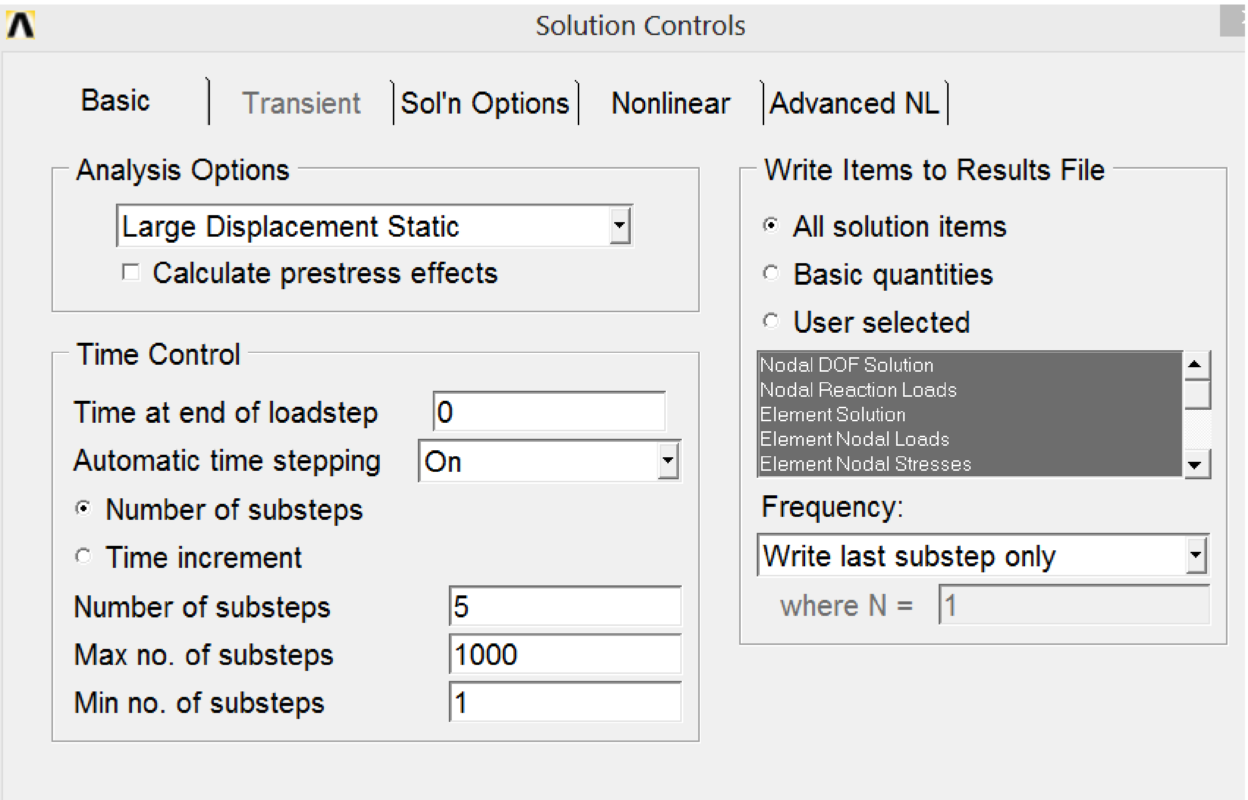

Fig. 3.3 Values that are fed in Sol'n n Control for non-linear analysis of Clamped-Clamped Beam

11.Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Keypoint 1 and Keypoint 3was fixed by making their displacement value to zero (ie all DOFs constrained).

12.Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Place different values of loads on Middle of the beam i.e Keypoint 2.

13.Solution > Solve > Current LS

14.GeneralPostproc->Plot Results->Counter Plot->Nodal solution->DOF Solution->Y-component of Displacement->OK

Procedure for performing non-linear static analysis on a Tapered Cantilever Beam (Point Load)

1.Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Beam->2 Node 188 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (2e05,0.3 respectively).

4.Preprocessor->Sections->Beams->Common sections->define breadth and height of the section(B=40, H=20)->Name it as small->Apply

5.Change ID and Name to 2 and Big->Change the dimension to the larger end of the tapered beam(B=80,H=20)->Click Apply->OK

6. Select Preprocessor->Sections->Beams->Tapered sections->By XYZ Location->Name the section as tapered and change the X location equal to length of the beam(1000mm)

7.Preprocessor->Modelling->create->Keypoints->In Active CS->

Create 2 keypoints

Keypoints	Coordinates(x,y)
1	(0,0)
2	(1000,0)

8. Go to Preprocessor->Modelling->create->Lines->Lines->Straight Lines->click on the two keypoints and OK.

9. Go to Preprocessor->Meshing->size control->Manual size->Global->size->set no. of element division to 50

10. Again Go to Preprocessor->Meshing->Mesh Attributes->Default attribs->set section no. to 3 tapered.

11. Preprocessor->Meshing->Mesh->Lines->click on line and OK

12.PlotControl->style->Size and shape->ON the display of element

13. Go to Solution ->analysis Type ->New analysis->select Static

14. Select Solution > Analysis Type >Sol'n Control...

The following values were fed as shown in fig.

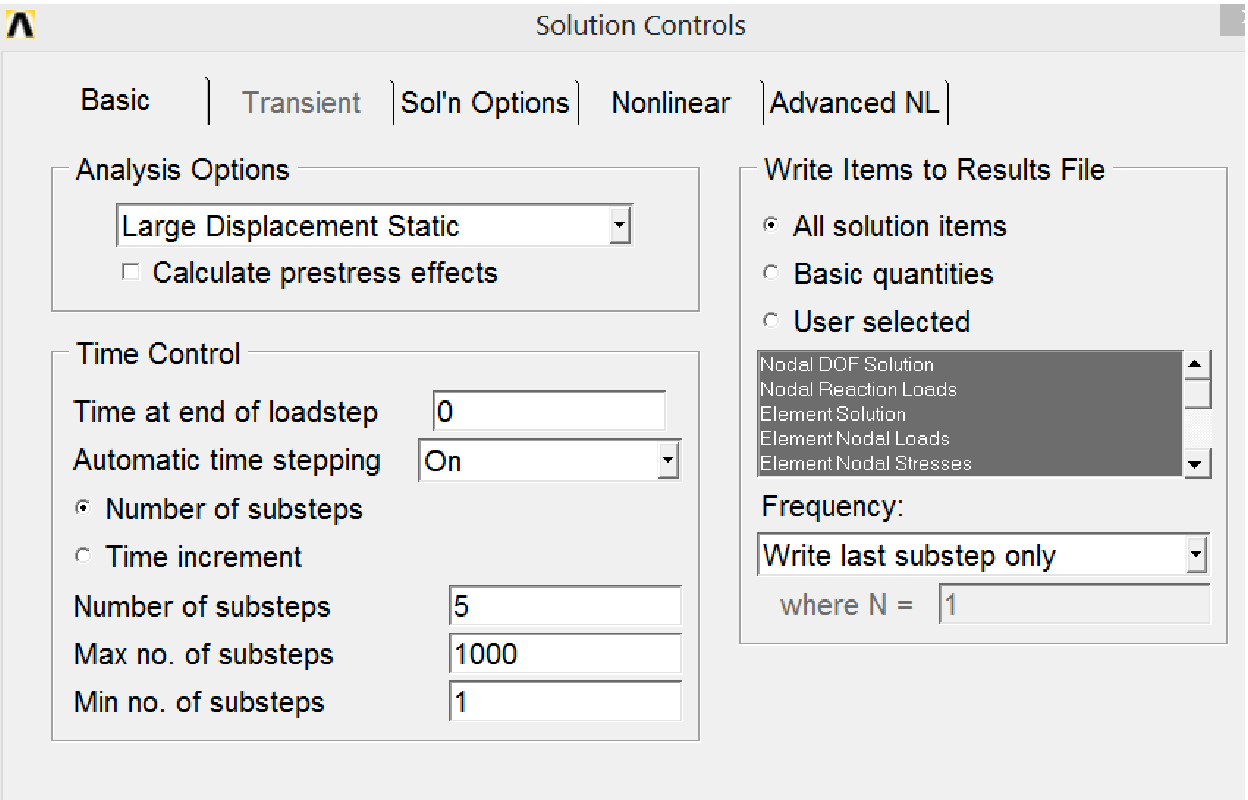

Fig. 3.4 Values that are fed in Sol'n n Control in non-linear analysis of Tapered Cantilever Beam

15. Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Keypoint 1 was selected and the displacement value was set to zero.

16. Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Different values of loads were given at keypoint 2 i.e bigger tapered end of the beam.

17.Solution > Solve > Current LS

18.GeneralPostproc->Plot Results->Counter Plot->Nodal solution->DOF Solution->Y-component of Displacement->OK

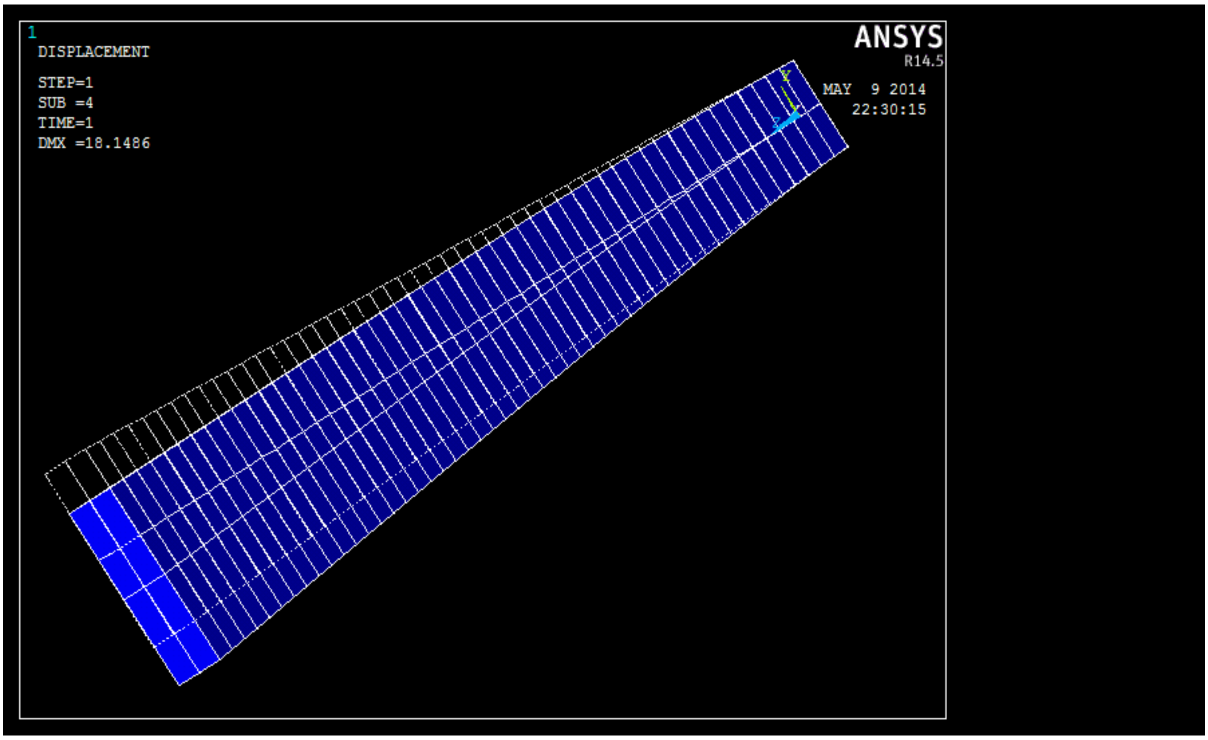

Fig. 3.5 Deflection Profile of a Tapered Cantilever Beam

Procedure for performing Pre-stressed Modal Analysis of a Tapered Cantilever Beam(Point Load)

1. Go to Preferences->select structural

2. Then go to Pre-processor->Element Type ->Add/Edit->Select Beam->2 Node 188 from the list of the elements.

3. Then go to Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic->
values of EX and PRXY were given (2e05,0.3 respectively).

4.Preprocessor->Sections->Beams->Common sections->define breadth and height of the section(B=40, H=20)->Name it as small->Apply

5.Change ID and Name to 2 and Big->Change the dimension to the larger end of the tapered beam(B=80,H=20)->Click Apply->OK

6. Select Preprocessor->Sections->Beams->Tapered sections->By XYZ Location->Name the section as tapered and change the X location equal to length of the beam(1000mm)

7.Preprocessor->Modelling->create->Keypoints->In Active CS->

Create 2 keypoints

Keypoints	Coordinates(x,y)
1	(0,0)
2	(1000,0)

8. Go to Preprocessor->Modelling->create->Lines->Lines->Straight Lines->click on the two keypoints and OK.

9. Go to Preprocessor->Meshing->size control->Manual size->Global->size->set no. of element division to 50

10. Again Go to Preprocessor->Meshing->Mesh Attributes->Default attribs->set section no. to 3 tapered.

11. Preprocessor->Meshing->Mesh->Lines->click on line and OK

12.PlotControl->style->Size and shape->ON the display of element

13. Go to Solution ->analysis Type ->New analysis->select Static

14. Select Solution > Analysis Type >Sol'n Control...

The following values were fed as shown in fig.

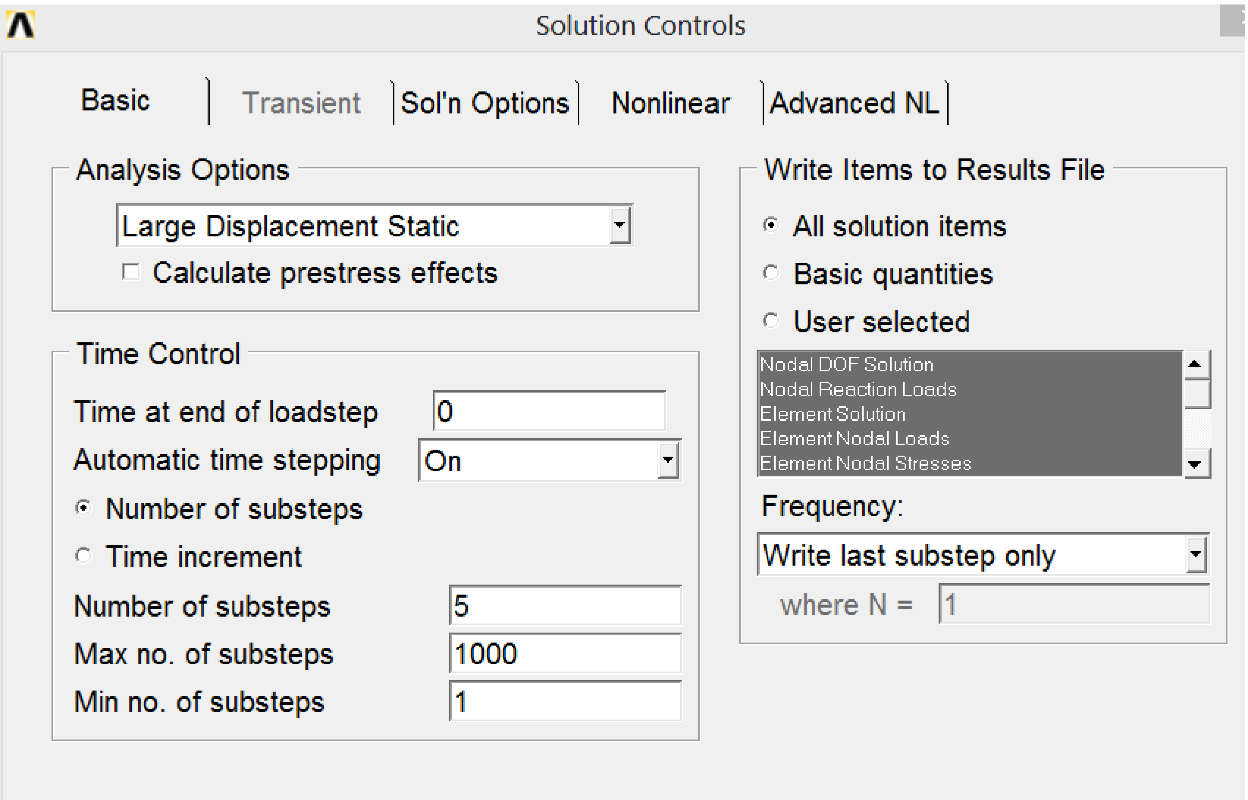

Fig. 3.6 Values thar are fed for non-linear analysis of Tapered cantilever beam

15. Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Keypoint 1 was selected and the displacement value was set to zero.

16. Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Different values of loads were given at keypoint 2 i.e bigger tapered end of the beam.

17.Solution > Solve > Current LS

18.GeneralPostproc->Plot Results->Counter Plot->Nodal solution->DOF Solution->Y-component of Displacement->OK

19.Save the file as .emat file.

20.Go to Solution->Analysis type->Select Modal

21. Go to Solution ->analysis Type ->Analysis option->select "Block Lanczos"

Next set the no. of modes to extract to 5

next click on "expand mode shapes" ,

Tick on "Include Prestress Effect"

set the frequency range from 0 to 10000

22.Go to Solution->Load step Opts->Other->Updeate Node coordinate->set Factor multiply disp. to 1->set ON "Key set disp. To zero"

23.Solve the analysis using Partial solve option "PSOLVE"

24. Generalpostproc->Result summary (list of all nodal frequencies are displayed)

25. Generalpostproc->Read Result->By Pick (picking the last nodal frequency)

26. General postproc->Plot Result->Deformed Shape->click on "deformed+undeformed shape"

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Frequency Response of Cantilever Beam in Modal Analysis.

E=71.72GPa, Poisson's ratio=0.33, Density=7850,Length=0.406m Breadth=0.02mHeight=0.002m

Result for the modal analysis performed on the cantilever beam in the previous chapter is provided in tabular form.

Set	Frequency (Hz.)	Result Generated from Numerical Prog. [9]	Load Step	Substep	Cumulative
1	4.1842	4.1630	1	1	1
2	16.717	-	1	2	2
3	26.220	26.1795	1	3	3
4	73.411	73.2710	1	4	4
5	104.57	-	1	5	5

Table 4.1 Frequency response of Cantilever Beam in Various Modes

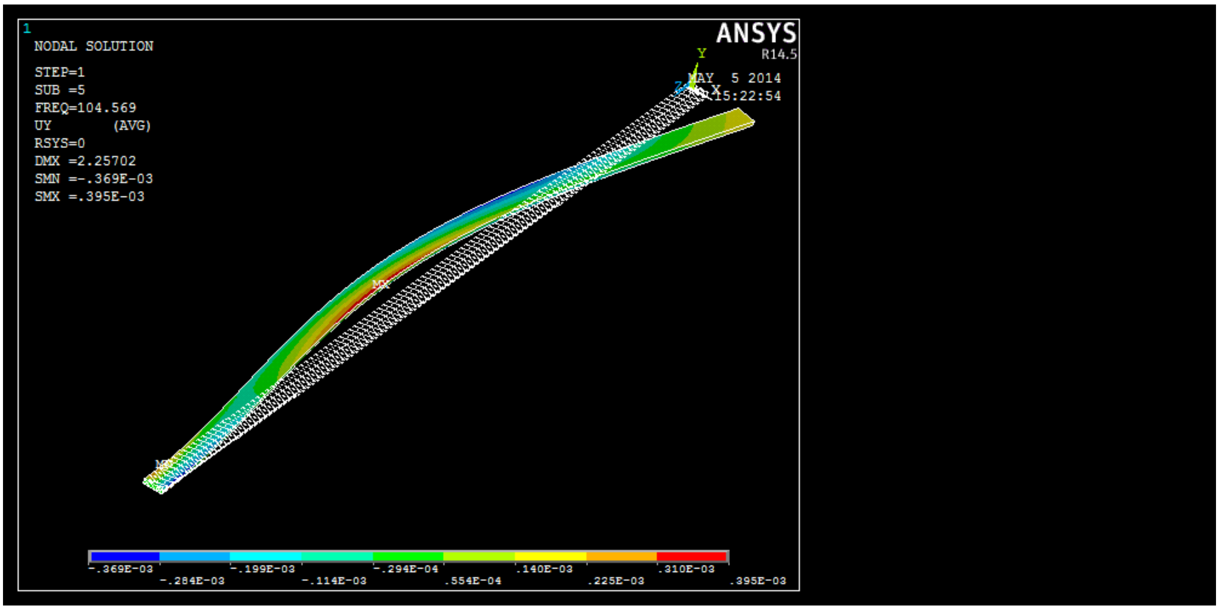

Fig. 4.1 Nodal Diagram of a Cantilver Beam Corresponding to n=5

4.2 Frequency response of Point loaded Clamped-Clamped Beam in modal analysis.

E=74.76 GPa ,Poisson's ratio=0.33 , Breadth=0.02m Height=0.005m Length=1m

Result for the modal analysis performed on the clamped-clamped beam in the previous chapter is provided in tabular form.

Set	Time/Freq	Result Generated from Numerical Prog. [9]	Load Step	Substep	Cumulative
1	26.665	26.5757	1	1	1
2	73.494	73.2074	1	2	2
3	106.19	-	1	3	3
4	144.06	143.3654	1	4	4
5	238.10	236.0882	1	5	5

Table4.2 Frequency response of Clamped-Clamped Beam in Various Modes

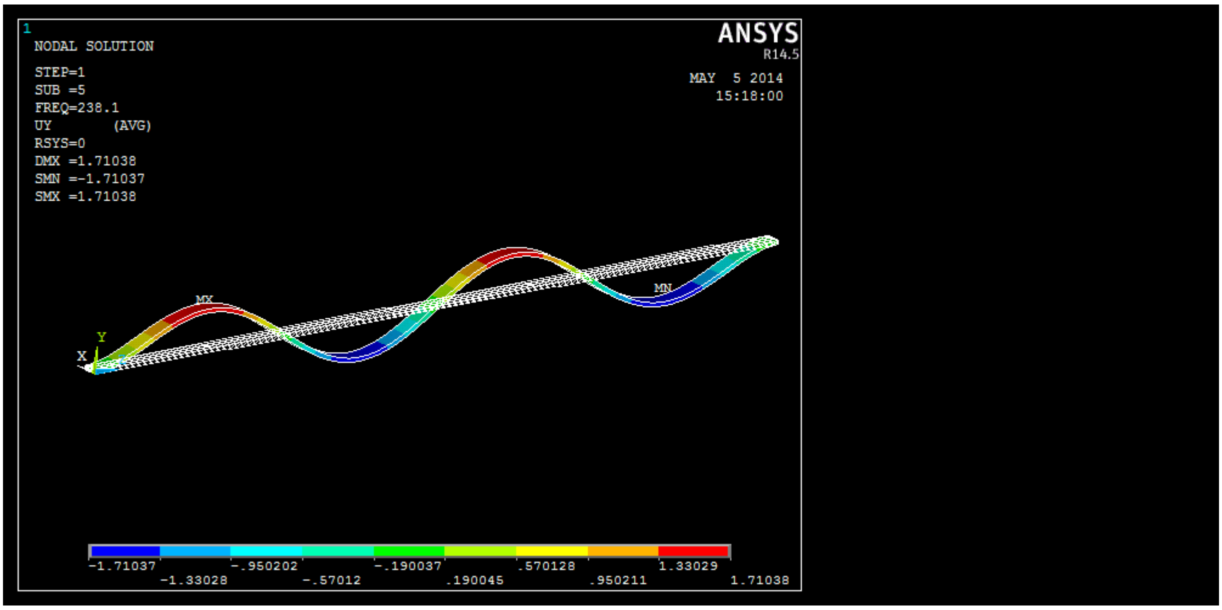

Fig. 4.2 Nodal Diagram of a Clamped-Clamped Beam Corresponding to n=5

4.3 Frequency response of Point loaded cantilever tapered beam in modal Analysis

E=200GPa Poisson's ratio=0.33 Density=2800 larger fixed cross-section=.08x.02m^2 smaller cross-section=0.04x0.02m^2 Length=1m

Result for the modal analysis performed on the clamped-clamped beam in the previous chapter is provided in tabular form.

Set	Time/Freq	Result Generated from Numerical Prog. [9]	Load Step	Substep	Cumulative
1	8.8888	8.995	1	1	1
2	42.602	42.5819	1	3	3
3	109.20	109.8564	1	4	4

4	210.2475	110.07	1	5	5

Table 4.3 Frequency response of Tapered Cantilever Beam in Various Modes

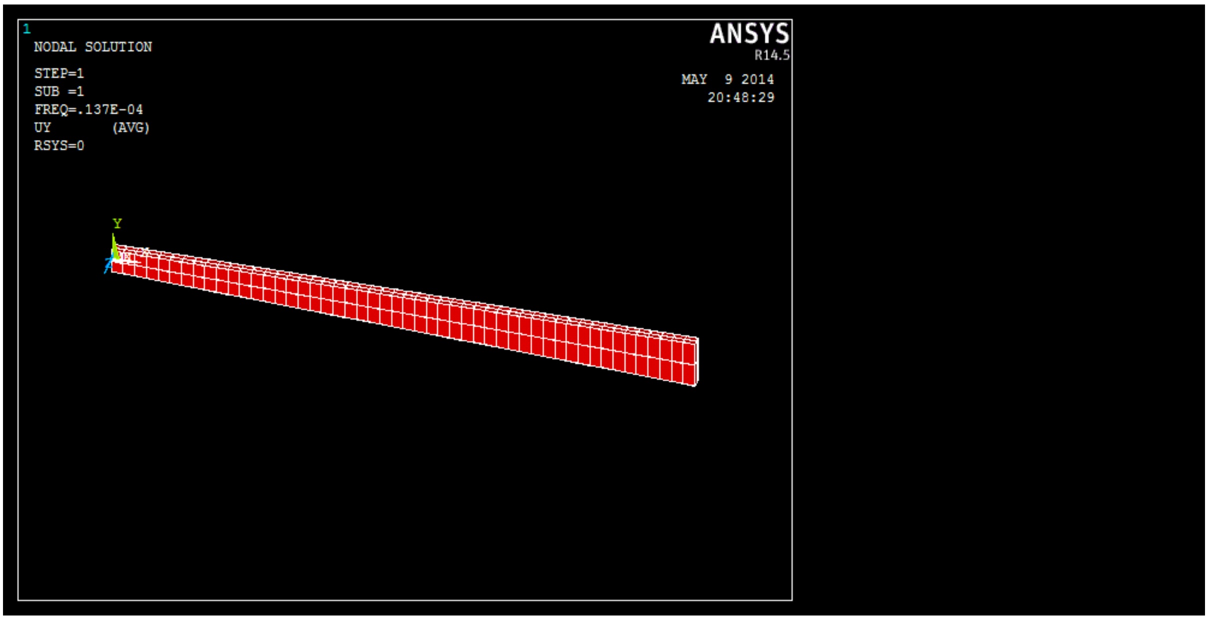

Fig. 4.3 Nodal diagram of a Tapered Cantilever Beam Corresponding to n=5

4.4 Behaviour of Point loaded Cantilever Beam in Non-Linear Static Analysis.

E=74.76GPa Density=2800 N/m^2 Poisson's ratio=0.33 Length=0.406m Breadth=.02m Height=0.002m

FORCE(Newton)	DISPLACEMENT(Metre)
0	0

20	0.004674
50	0.011679
100	0.023326
150	0.034841
200	0.046234
250	0.057447
300	0.068438
350	0.079183
400	0.089655
450	0.099834
500	0.109708
700	0.145971
1000	0.191037
1500	0.246226
2000	0.284678
2500	0.312827
3000	0.334353

Table 4.4 Deflection Corresponding to various forces in non-linear Static analysis of Cantilever Beam

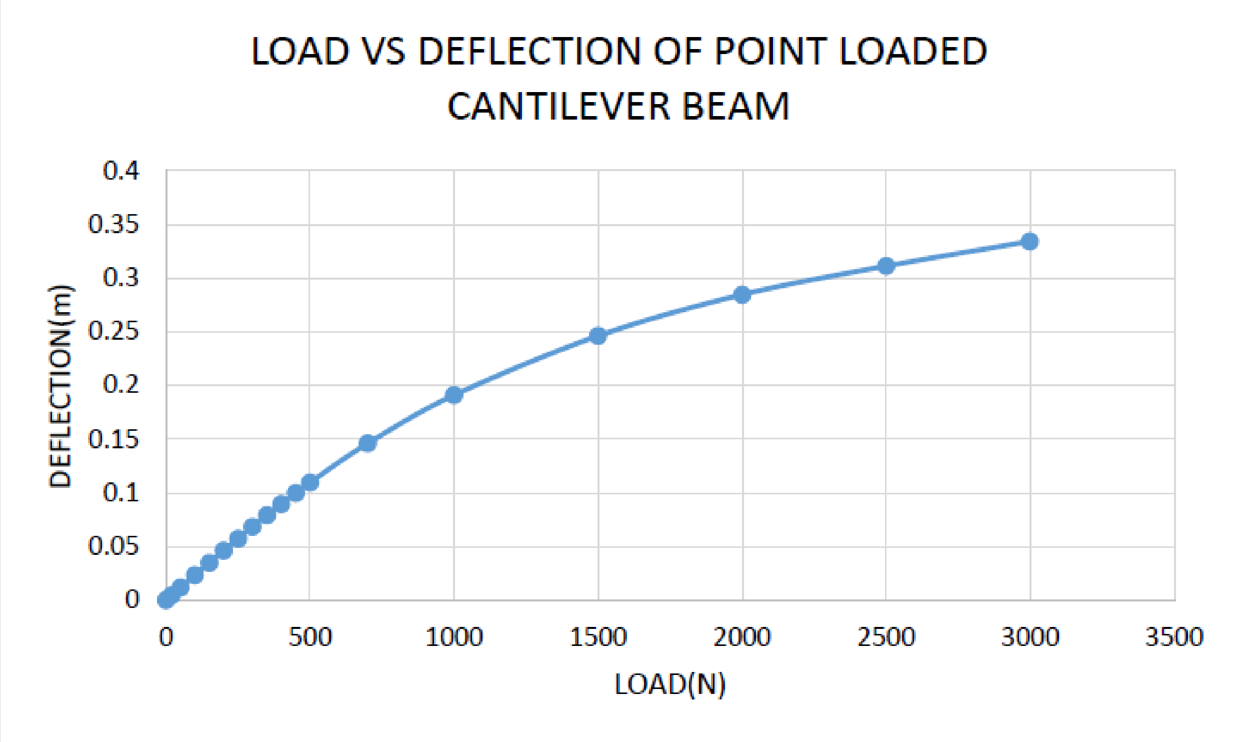

Fig. 4.4 Load vs Deflection of Point Loaded Cantilever Beam

4.5 Behaviour of Point loaded Clamped-Clamped Beam in Non-Linear Static Analysis.

E=74.76GPa Density=2800 N/m^2 Poisson's ratio=0.3 Length=1m, Breadth=0.02, Height=0.005.

Load(Newton)	Displacement(meter)
0	0
2	0.000042
6	0.000126
10	0.00021
30	0.00063
50	0.001048
100	0.002048
300	0.005927

500	0.009146
700	0.011812
1000	0.015059
1500	0.01924
2000	0.022509
2500	0.025228
3000	0.027576

Table 4.5 Deflection Corresponding to various forces in non-linear Static analysis of Clamped-Clamped Beam

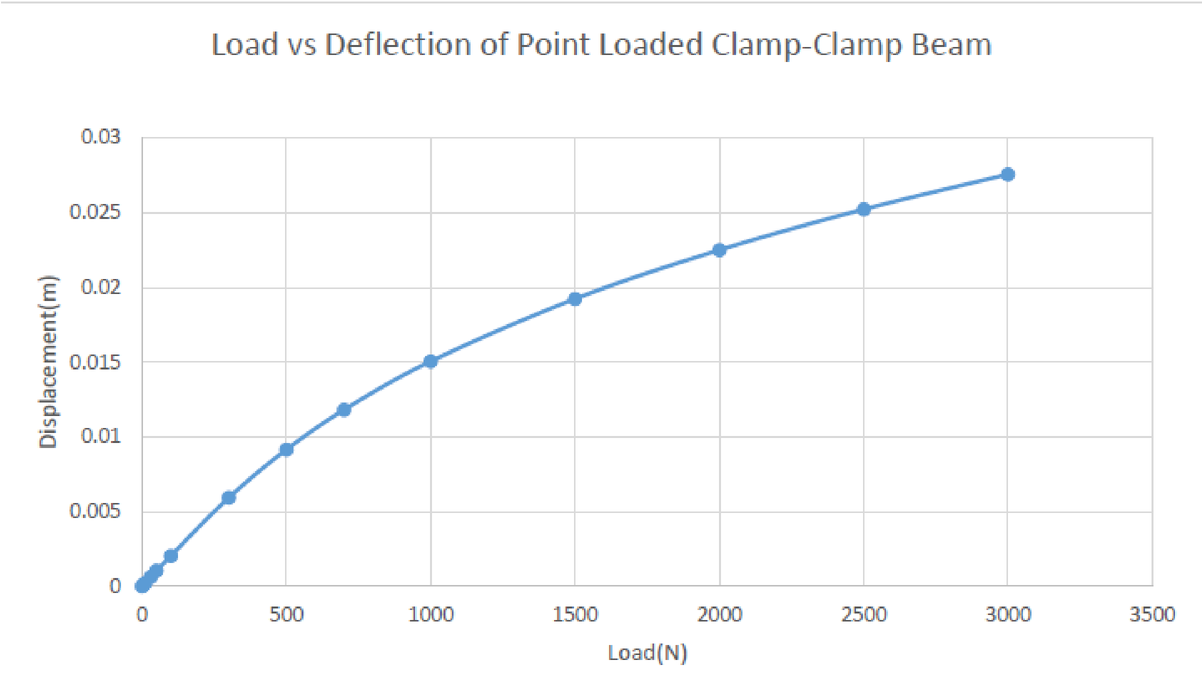

Fig. 4.5 Load vs Deflection of Point Loaded Clamp-Clamp Beam

Behaviour of Point loaded Tapered Cantilever Beam in Non-Linear Static Analysis.

E=200GPa, Posson's ration=0.3,Density=2800 Tapered cross-section smaller, section=.02x.008 Bigger section=.02x.01

Load(Newton)	Deflection(meter)
0	0

5	0.001318
25	0.006592
50	0.013182
100	0.026357
300	0.078681
500	0.129914
700	0.179444
1000	0.249541
1500	0.353314
2000	0.440597
2500	0.513321
3000	0.574076
4000	0.668839
5000	0.738967

Table 4.6 Deflection Corresponding to various forces in non-linear Static analysis of Tapered Cantilever Beam

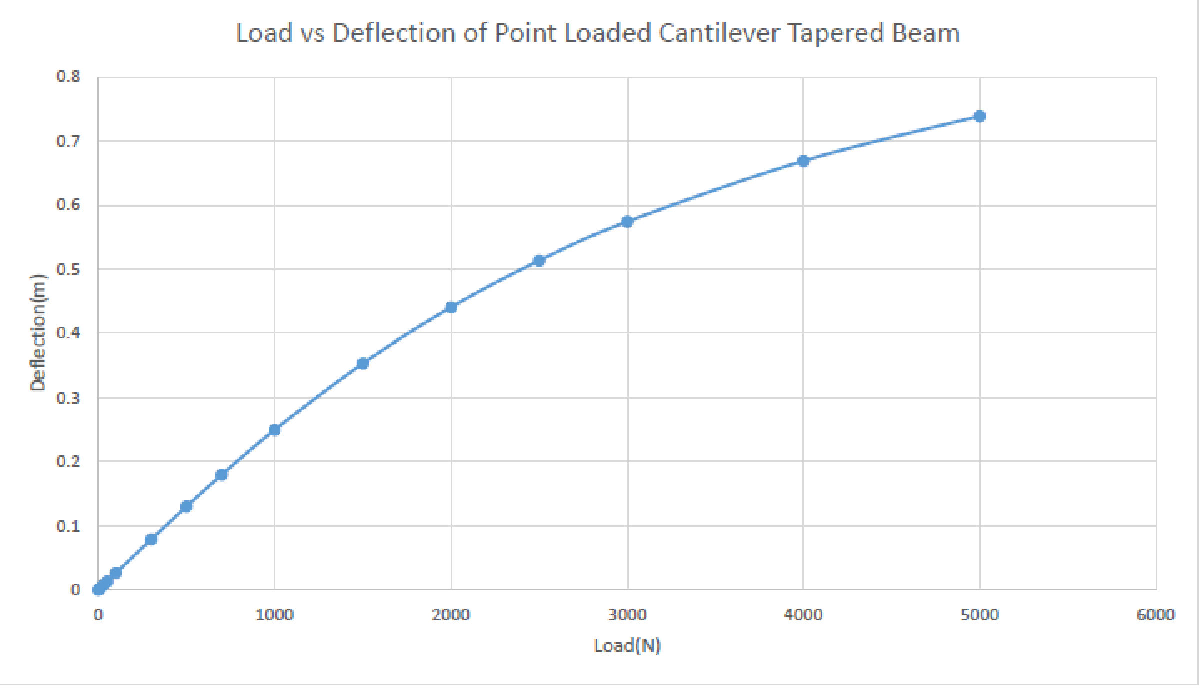

Fig. 4.6 Load vs Deflection of Point Loaded Tapered Cantilever Beam

Pre-stressed Modal Analysis of Tapered Cantilever Beam

L = 1m, b = 0.02m, t = 0.01m (Larger Section, fixed/root); b = 0.02m, t = 0.005m.

E = 71.7GPa, Poisson's ratio = 0.3, Density = 2800kg/m3.

Force	Frequency
5	8.8993
10	8.9127
30	9.038
50	9.2676
70	9.5674
100	10.082
120	10.439
150	10.968
200	11.796

Table 4.7 Frequency Response of Tapered Cantilever Beam in Pre-stressed Condition

CHAPTER 5

CONCLUSION and FUTURE SCOPE

CONCLUSION

The objective of the present thesis work was to study the pre-stressed modal analysis of tapered beams and therefore determine the backbone curves of the system through ANSYS. In the course of the study, modal analysis was performed for uniform and non-uniform beams under two different boundary conditions (clamped-free and clamped-clamped). A large displacement static analysis (incorporating the effect of geometric nonlinearity) was also performed separately. The generated results clearly indicated nonlinear behaviour at higher load range. Finally, a procedure for prestressed modal analysis for beams was developed, where a nonlinear static analysis is performed first, followed by a modal analysis that includes the effect of the prestress. The results for Pre-stressed modal analysis of a tapered cantilever beam was generated which validated the correct trend.

FUTURE SCOPE OF STUDY

1. The material considered is homogenous and isotropic. For further research the material non-linearity can be taken into account.
2. In the modern industrial application context, functionally graded materials are very important. Future work can include both depth-wise and length-wise gradation of materials for dynamic analysis of beams.
3. The project is done on beams of uniform and linearly tapered thickness. Further work can be done on beams with exponential or parabolic variation of thickness. It is also possible to consider, variation of width separately or along with variation of thickness.

REFERENCES

[1]. M. Baghani, H. Mazaheri, H. Salarieh . "Analysis of large amplitude free vibrations of clamped tapered beams on a nonlinear elastic foundation". Applied Mathematical Modelling. 38, 1 February 2014, 1176-1186.

[2]. A. Allahverdizadeh, I. Eshraghi, M.J. Mahjoob, N. Nasrollahzadeh. "Nonlinear vibration analysis of FGER sandwich beams." International Journal of Mechanical Sciences, 78, January 2014, 167-176.

[3]. H. Arvin, F. Bakhtiari-Nejad. "Non-linear modal analysis of a rotating beam." International Journal of Non-Linear Mechanics, 46, July 2011, 877-897

[4]. Lianhua Wang, Jianjun Ma, JianPeng, Lifeng Li. "Large amplitude vibration and parametric instability of inextensional beams on the elastic foundation". International Journal of Mechanical Sciences, 67, February 2013, 1-9.

[5]. M. Baghani, R.A. Jafari-Talookolaei, H. Salarieh. "Large amplitudes free vibrations and post-buckling analysis of unsymmetrically laminated composite beams on nonlinear elastic foundation". Applied Mathematical Modelling, 35, January 2011, 130-138.

[6]. Carlos E.N. Mazzilli, César T. Sanches, Odulpho G.P. BarachoNeto, Marian Wiercigroch, Marko Keber. "Non-linear modal analysis for beams subjected to axial loads: Analytical and finite-element solutions". International Journal of Non-Linear Mechanics, 43, July 2008, 551-561.

[7]. SandroCarbonari, FabrizioGara, DavideRoia, GrazianoLeoni, LuiginoDezi. "Tests on two 18-years-old prestressed thin walled roof elements." Engineering Structures, 49, April 2013, 936-946.

[8]. T. Kocatürk, M. Şimşek. "Dynamic analysis of eccentrically prestressed viscoelastic Timoshenko beams under a moving harmonic load." Computers & Structures, 84, December 2006, 2113-2127.

[9]. S. Kumar and A. Mitra, "Large amplitude free vibration analysis of tapered beams under different boundary conditions". Proceedings of 10th International Conference on Mechanical Engineering, ICME 2013, Dhaka, Bangladesh. (Accepted)